AIAA 98-0924
Numerical Investigation of Twin-Nozzle Rocket Plume Phenomenology, Part II

Houshang Ebrahimi
Sverdrup Technology, Inc., AEDC Group
Arnold Engineering Development Center
Arnold Air Force Base, Tennessee

36th Aerospace Sciences Meeting& Exhibit
January 12 - 15, 1998 / Reno, NV

Numerical Investigation of Twin-Nozzle Rocket Plume Phenomenology, Part II*

*Houshang B. Ebrahimi***
Sverdrup Technology, Inc., AEDC Group
Arnold Engineering Development Center
Arnold Air Force Base, TN 37389-9013

Abstract

The Generalized Implicit Flow Solver (GIFS) computer program has been modified and applied for analysis of three-dimensional, reacting, two-phase flow simulation problems. The intent of the original GIFS development effort was to provide the Joint Army, Navy, NASA, Air Force (JANNAF) community with a standard computational methodology to simulate multiple nozzle/plume flow-field phenomena and other three-dimensional effects. The Van Leer Flux Splitting option has been successfully implemented into the existing GIFS model and provides a more robust solution scheme, making application of the model more reasonable for engineering applications.

This paper is a continuation of the previous work and reports the significant results of parametric flow-field simulations resulting from a dual-nozzle propulsion system operating at a high-altitude flight condition. In support of this effort, four calculations of Titan II SLV flow fields have been completed to assess the effects of three-dimensionality, missile body effects, chemistry, and gas generator flow on simulated plume exhaust flow-field properties.

These calculations indicate that three-dimensionality is always an important factor and could substantially influence the interpretation of the results. If three dimensional effects are oversimplified in the model, analyses of the spatial results can be misinterpreted and misapplied. The missile body effect can also generate three-dimensional influences affecting the Mach reflection location, the plume/plume impingement shock location, inviscid shock structure, and shear layer growth. The gas generator flow influences the very near-field simulation but is significantly dampened beyond approximately 1 meter downstream of the nozzle exit (nonreacting). If missile base heating, recirculation effects, or nozzle impingement heat-transfer analyses are required, the gas generator flow, including a kinetic chemistry model is the dominant influence.

Introduction

In order to support propulsion testing and analysis requirements of the aerospace and exhaust plume community, a need exists for a fluid dynamics model that solves the fully coupled two-phase Navier-Stokes equations in multiple dimensions. Evaluations of solid-propellant rocket motor performance, nozzle erosion, and rocket plume radiative transfer analyses require a computer model that simulates complex three-dimensional, chemically reacting, two-phase flow effects.[1] Although this type of full Navier-Stokes method provides an accurate, qualitative description of the basic features of the propulsion-generated flow fields, quantitative simulations for predicting fundamental parameters such as base pressure and heat transfer, gas static pressure, temperature, and chemical composition in the flow-field domain have not been validated. In the past few years, significant progress has been made in the areas of numerical rocket flow simulations and computational resources to the point that Navier-Stokes solutions are viable analysis tools. Efforts to validate these models tend to lag the algorithm development and the computational hardware advancements considerably and hamper the overall confidence of the quantitative results. This study provides a qualitative validation of the physical effects included in the GIFS model and provides a basis for in-depth understanding of the results of the GIFS model.

* The research reported herein was performed by the Arnold Engineering Development Center (AEDC), Air Force Materiel Command. Work and analysis for this research were performed by personnel of Sverdrup Technology, Inc., AEDC Group, technical services contractor for AEDC. Further reproduction is authorized to satisfy needs of the U. S. Government.

** Senior Member, AIAA.

Approved for public release; distribution unlimited.

The flow fields generated by rocket propulsion systems are complex, with regions of strong inviscid/viscous interactions, free-stream shear layers, nozzle wall and missile body boundary layers, external and internal shocks, separation regions, and plume/plume impingement and associated flow interactions for multiple nozzle designs, all with chemically reacting kinetics.[2] Coupling all of these phenomena simultaneously in a numerical simulation tool challenges the state of the art (SOA) for CFD models. To account for all phenomena affecting plume flow properties and the resulting radiative transfer implications, computationally efficient, multidimensional computer models are required.

Recenty there has been increased emphasis on the application of CFD models, both in the commercial arena and government-developed computer programs, to simulate complex multidimensional plume phenomena. A large majority of the solutions obtained to date are based on the perfect gas (constant gamma) approximation, as fully reacting flows are considerably more complex and difficult to solve. Including the effects of chemistry in the solution produces a stiff set of equations that are numerically difficult to solve using conventional algorithms. In addition, the grid resolution requirements become more severe, and time step issues arise when reacting chemistry is included in the Navier-Stokes model.

Conventional rocket exhaust flow computer models[3,4] employ Euler solutions for the plume core flow and superimpose a turbulent parabolic mixing approach for the free-stream air and plume entrainment (shear) layer. These methods are generally not adequate for situations when the flow is not fully dominated by either inviscid core expansion or the plume afterburning phenomena. Three-dimensional features produced by multiple nozzle propulsion systems and vehicle body/base interactions cannot be sufficiently treated through the use of these models. Inaccurate accounting of the 3D upstream influences on the plume shear layer development and the resulting plume structure is one of the primary issues preventing approximate models from accurately simulating the overall flow-field phenomena.[5]

The GIFS numerical algorithm provides a solution of the two- and three-dimensional Reynolds-averaged Navier-Stokes (NS) equations using the MacCormack implicit finite-volume algorithm with Gauss-Seidel line relaxation.[6] Several 2-D and 3-D plume flow-field calculations have been completed for the plume near-field region using the original version of the GIFS model.[6] The GIFS model includes a frozen and generalized finite-rate kinetic chemistry model, a Lagrangian particle model for treating solid and liquid particulates, and a two-equation turbulence model, as well as a laminar model. These complex phenomena are required to accurately simulate the physics expected to contribute to the flow-field spatial distribution of gas dynamic, thermodynamic, and chemical properties. The Van Leer Flux Splitting option[7] has been successfully implemented into the original GIFS model and provides a more robust solution scheme for simulating propulsion flow-field phenomena.[8] This enhanced version of the GIFS model was applied in this study.

The near-field plume flow field emanating from a multinozzle vehicle flying at high altitude are largely dominated by the processes taking place in the plumes' interaction region. Prior to the release of the GIFS model, simulations of multiple nozzle/plume flow fields were commonly treated by assuming a single equivalent nozzle configuration having equal mass, energy, and momentum of the multiple nozzle geometry. Further, uniform (one-dimensional) nozzle exit flow properties were used as the starting conditions for the plume calculation as a simplifying assumption. The simplified model assumes that the details of the 3-D flow structure in the near-field flow and the 2-D start condition are unimportant and that the flow processes affecting the plume shear layer initialization (such as base separation and recirculation) will be dominated by the overall ambient flow entrainment effects. For spatial analysis requirements the level of agreement between computations based on the single equivalent nozzle methodology and simulations from multiple nozzle propulsion systems has not been acceptable. The source of the disagreement is due, at least in part, to an incorrect physical model of the phenomena dominating the observations, e.g., inadequate turbulence, incomplete chemical mechanisms, missing or inaccurate reaction rates, simplified initial start conditions and three-dimensional geometry effects. The actual physical geometries are often oversimplified, and relevant details of the engine design and the resulting impact on the initial conditions and the downstream exhaust flow are frequently ignored.

In the previous study,[5] six simulations of the Titan II Plume flow field were completed to assess the effects of three dimensionality, turbulent viscosity, finite-rate chemical kinetics, intranozzle geometric spacing, and nozzle exit profile initial conditions. This study is a continuation of the previous work and further demonstrates the significance of three-dimensional effects as applied to multiple-nozzle rocket missile plume simulations. The results can be applied to guide efforts to determine how these phenomena may be approximated to improve engineering approaches, and to explore how computer resource requirements for simulating three-dimensional solutions can be reduced. The results presented in this paper demonstrate continued progress in simulating the rocket exhaust flow field from a multiple-nozzle propulsion vehicle in flight.

Computations

This computational effort consisted of four, three-dimensional twin-nozzle calculations for the Titan II vehicle at actual flight conditions. The 2D-axisymmetric initial conditions were computed using the Two-Dimensional Computer Program, TDK (Ref. 9) and were identical for all cases. For all cases, the plume flow field was simulated at an altitude of approximately 50 km at a free-stream Mach number of 5.7, assuming turbulent conditions in all flow regions. The first case simulates three-dimensional, reacting flow conditions and includes flow regions surrounding the missile body, the base flow region, the gas generator flow, and the exhaust plume domain assuming reacting chemistry conditions. The second case is similar to the first, assuming frozen flow conditions. To assess the effects of the gas generator, the second case was repeated without consideration of the gas generator in the model. The final solution assumed a single equivalent axisymmetric nozzle to approximate the three-dimensional solution geometry. Included in Table 1 is a summary of all the parametric flow-field conditions and assumptions evaluated in this study and the previous investigation. Cases 1-6 were reported in the previous paper.[5] This study focused on the last four cases.[7-10]

The Titan II Space-Launched Vehicle (SLV) propulsion system is propelled by two engines located on either side of a plane of symmetry passing through the vehicle centerline. Given in Fig. 1 is a schematic of the vehicle. The sea-level rated thrust derived from each engine is approximately 214,000 lbf. Roll, pitch, and yaw control are pro-

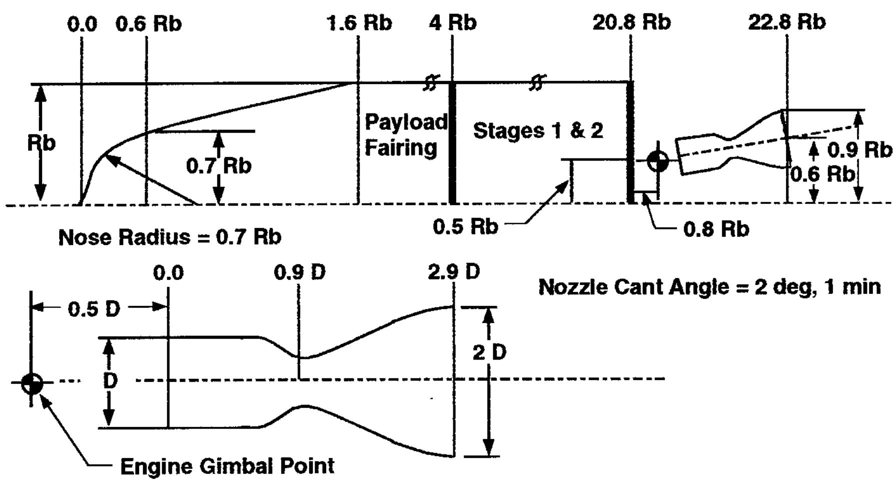


Fig. 1. Vehicle dimensions.

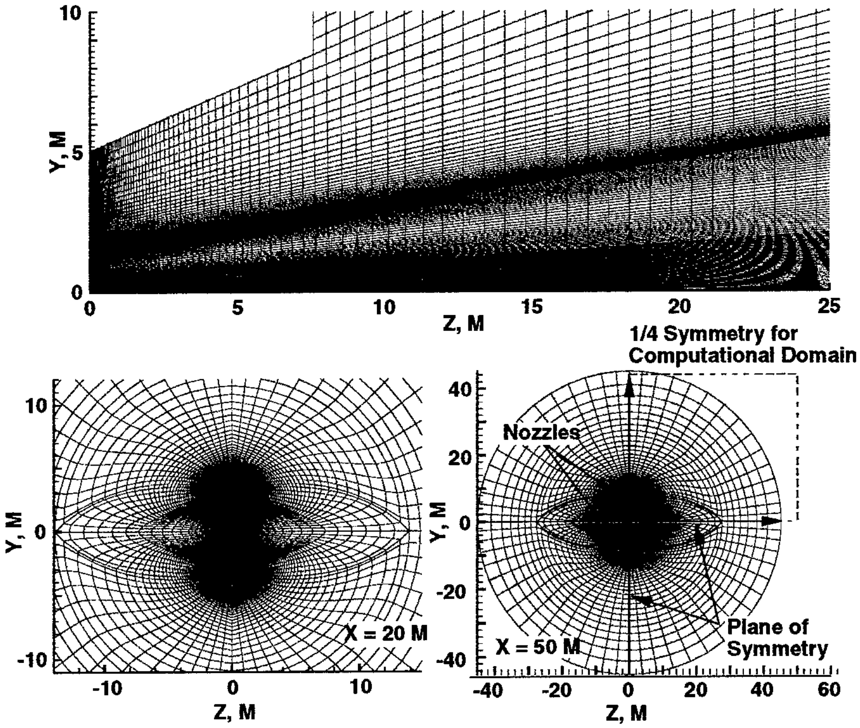
Table 1. Parametric Flow-field Conditions

Case	Viscosity	Inlet Profile	Chemistry	Nozzle Spacing	GG
1	Laminar	1-D	Constant γ	Wide	no
2	Turbulent	1-D	Constant γ	Wide	no
3	Turbulent	1-D	Constant γ	Narrow	no
4	Turbulent	1-D	Finite Rate	Narrow	no
5	Turbulent	1-D	Frozen	Narrow	no
6	Turbulent	2-D	Finite Rate	Narrow	no
7	Turbulent	2-D	Finite Rate	Narrow	yes
8	Turbulent	2-D	Frozen	Narrow	yes
9	Turbulent	2-D	Frozen	Narrow	no
10	Turbulent	2-D	Finite Rate	2D-axisymmetric	no

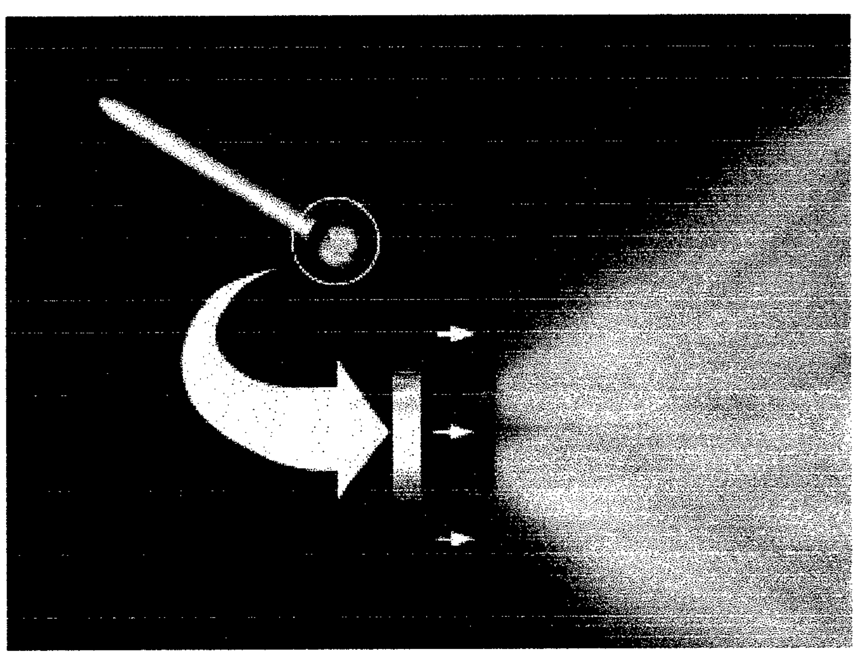
vided by gimballing of the engine 5 deg from the engine neutral position (2-deg cant angle away from the vehicle centerline for both nozzles). The 2-deg cant angle was included in the three-dimensional simulations.[10] Both nozzles are identical in geometry and operating conditions.

The thrust chamber assemblies and the nozzle skirt are regeneratively cooled. In addition, injector spray patterns intentionally direct a fuel-rich layer adjacent to the chamber walls to reduce the wall heat loads. Power to drive the turbopumps is derived from two gas generators which represent approximately 1.5 percent of the overall propellant expended by both engines. These gas generators are intentionally operated at fuel-rich conditions to minimize heat loads on the turbine blades.

To limit the number of grid points and minimize the CPU execution time, quarter-plane symmetry assumptions were made for the computational domain. All simulations were accomplished at zero angle of attack. A total of 4 million grid points was utilized in the computational domain for the three-dimensional simulations, and 200,000 grid points for the axisymmetric case. The exhaust plume portion of the computational domain is shown in Fig. 2a, with the orientation indicated by the axes. A schematic of the 3-D plume configuration is shown in Fig. 2b. The external airflow conditions and nominal 1-D liquid-propellant rocket nozzle throat and exit conditions and



a. Schematic of the 3-D grid and the planes of symmetry



b. Schematic of the 3-D plume

Fig. 2. Geometry representation of the computation domain.

gas generator conditions for the Titan cases are presented in Table 2. The calculations assuming finite-rate chemical kinetics used a chemical reaction model consisting of 10 species and 11 reactions for carbon, hydrogen, oxygen, and nitrogen-based propellant systems. The κ-e turbulence model was used for the viscous stress approximation. The computation was performed on a single-processor SGI Power Challenge R8000. For the three-dimensional cases, the calculations required approximately twelve weeks of CPU time to converge. In contrast, the axisymmetric case required 5 days to converge. The GASP computer program11 contains a hybrid Navier Stokes/ Parabolized Navier Stokes (PNS) methodology which was also applied for these calculations. The results compared favorably with the GIFS calculations; however, the computational time was significantly reduced by using the PNS method in the supersonic flow-field regions. For these cases, the 3D calculation time was reducted to four weeks and the axisymmetric to 18 hours using the PNS model.

Table 2. Inflow Conditions

Ambient Conditions at 47.6 km

- Tinf = 269 K
- V = 1877.6 m/sec
- Cp/Cv = 1.4
- r = 1.388×10^{-3} kg/m^3
- P = 108 Pa
- Mach = 5.7
- Species Concentrations (Mass Fraction)
  N_2 = 0.77
  O_2 = 0.23

Jet Conditions (One Dimensional)

- T = 1920 K
- V = 2776.6 m/sec
- P = 92800 Pa
- r = 1.68×10^{-1} kg/m^3
- Mach = 3.0
- Species Concentrations (Mass Fraction)

CO = 0.039	CO_2 = 0.1811	H_2O = 0.3496	
N_2 = 0.414	NO = 0.0109	OH = 2.139e-3	
H_2 = 3.13e-3	H = 1.24e-4	O_2 = 0.0	O = 0.0

Gas Generator Conditions

– M = 1.01	– CH_4 = 0.135
– T = 899 K	– H_2 = 0.035
– P = 85488.9 Pa	– H_2O = 0.034
– CO = 0.04	– NH_3 = 0.253
– CO_2 = 0.004	– C = 0.038

Including the previous work, ten numerical simulations of the Titan II twin-nozzle configuration flow field were obtained at Mach 5.7. Different combinations of flow assumptions and approximations were applied in an attempt to isolate individual physical influences and effects. In the previous study, only the exhaust plume flow field was computed. The upstream influence of the missile body, the missile base region, and the gas generator flow were not considered.

The numerous assumptions and specifics of the all the solutions are summarized briefly. Cases 1 and 2 incorporated laminar and turbulent viscous stress models, respectively, and were computed assuming a perfect gas equation of state. Case 3 was a turbulent, constant gamma approximation with an intranozzle geometric spacing that differed from Cases 1 and 2. Cases 4 and 5 differed in the chemistry approximation, assuming frozen and finite-rate chemistry, respectively. Cases 1-5 all assumed uniform (1-D) nozzle exit properties as the GIFS start line conditions. Case 6 assumed a 2D start line profile. Cases 1- 6 were all initiated at the nozzle exit location and did not include upstream body/base region in the simulation. The nozzle exit plane (calculated via TDK) was used to define the starting boundary condition for the GIFS plume calculation. Cases 7, 8, and 9 include upstream influences and differ in the chemistry assumption and inclusion of the gas generator in the simulation. Case 7 simulated chemically reacting flow over the missile body, in the base region, and the exhaust plume domains. The gas generator flow was included in this simulation. Case 8 contrasts the influence of the chemistry by repeating the previous calculation considering a frozen flow assumption. Case 9 is similar to Case 8, but excludes the gas generator flow. Case 10 is an axisymmetric single equivalent nozzle approximation of the twin-nozzle geometry and includes the missile body, missile base region, and exhaust plume in the computational domain. The gas generator flow was not considered in the axisymmetric simulation.

The following will highlight the effects evident from contrasting the various solutions.

Contrasting the turbulent and laminar solutions indicated that the barrel shock reflection point is located approximately 8 meters farther downstream for the turbulent solution. Therefore, the spatial characteristics of the inviscid plume flow field structure can be influenced by the viscous stress model included in the solution. In the present case, at 47.6-km altitude, the effect of turbulence is not particularly strong. However, at lower altitudes where higher Reynolds numbers are expected, turbulent effects will be more predominant.

The influence of intranozzle spacing was also investigated. The results indicate that intranozzle spacing has a noticeable influence on the flow-field structure. The initial plume expansion angle is larger for the wider spacing case, and the shock reflection location is farther downstream. It is concluded that the intranozzle spacing affects the plume impingement shock location, the inviscid shock structure, and the shear layer.

The effects of the chemistry model were also evaluated. Comparison of solutions assuming frozen and kinetic chemistry approximations indicates that the chemistry model influences somewhat the location of the barrel shock reflection point. The other features of the plume exhaust flow are not drastically affected by the chemistry assumption. However, these simulations were accomplished at high-altitude conditions, where the entrainment of the atmospheric gases into the plume shear layer does not result in significant afterburning. It is expected that chemistry will have a significant influence on solutions at lower altitudes where plume/ atmospheric afterburning will dominate the flow field. The chemical kinetic effect strongly influences the calculation in the immediate missile base region, especially if the fuel-rich gas generator flow is considered in the simulation.

The influence of two-dimensional versus one-dimensional startline conditions was also evaluated. A comparison of the spatial plume size indicates that the uniform starting line results in a slightly larger plume expansion region due to the increased pressure gradient between the nozzle exit and the free stream. As expected, differences between the two start line approximations become less significant as the flow progresses axially downstream. Although these two solutions do not show a significant effect of exit profile shape on the plume temperature, it is expected that in other cases, especially those with strong afterburning, larger effects would be noted.

It should also be noted that all solutions assumed constant oxidizer/fuel (O/F) ratio across the nozzle exit plane. When non-ideal engine effects such as fuel-film cooling and injector imperfections are accounted for, the O/F ratio and, consequently, the chemical composition of the exhaust products can vary widely across the nozzle exit plane, resulting in nonaxisymmetric chemical distributions. Prediction of the O/F distribution is beyond the scope of this study and outside of the current capabilities of the GIFS model.

Three-Dimensional Effect

In an earlier study by one of the authors[2] and others,[5,10] it was shown that three-dimensional effects are important and should not be ignored or oversimplified in modeling efforts. Previous work focused on three-dimensional plume exhaust phenomena but did not consider the upstream influence of the missile body, missile base region, or the gas generator flow. This study further investigated the three-dimensional aspects of CFD simulations, including upstream influences generated by the Titan II missile body, base region, and fuel-rich gas generator flow. Static pressure and Mach number contours resulting from these three-dimensional calculations initiated at the missile nose are shown in Figs. 3 and 4, respectively. Figure 4 includes Mach number cross sections taken at the 100-m

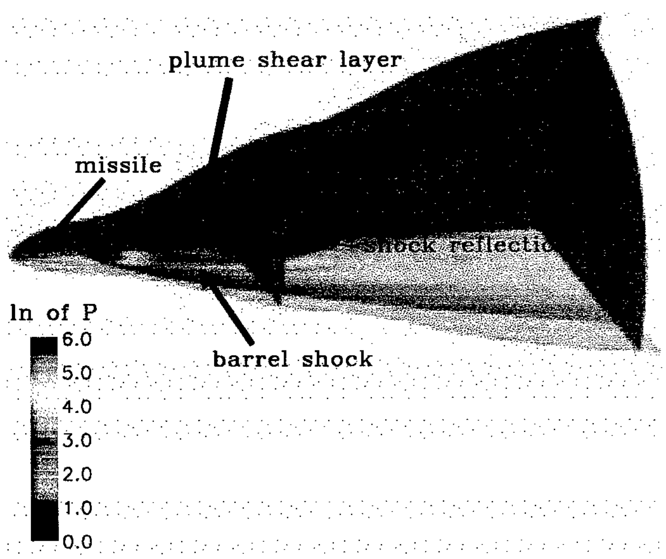


Fig. 3. Pressure contours (3-D calculations).

and 205-m axial locations as referenced to the nozzle exit. This simulation also assumesf finite, rate-controlled chemistry throughout the solution domain, including the fuel-rich gas generator flow region. The plume expansion shock (barrel shock) formation in the plume near-field and the shock reflection visible at 116 meters downstream of the nozzle exit plane location are evident in Figs. 3 and 4. The static temperature is increased immediately downstream of the reflection point to approximately half the value of the total temperature of the flow. The blunt body shock, plume expansion shock, barrel shock reflection and resulting separation region are clearly evident in these results. Figures 5a and 5b are close-up views of static temperature contours and Mach number vectors, respectively, in the missile base flow region. The heating of the missile base surface and the effect of the gas generator effluent impinging on the nozzle surfaces are clearly evident. Further, the gas generator flow initially expands as it exits the nozzle, adjusting to the ambient pressure condition, but further downstream in the region between the two nozzles, the gas generator flow is compressed because of the area change created by the nozzle expansion skirts. In Fig. 5b, showing the Mach vectors, the interaction of the gas generator flow with the base region and the nozzles is apparent. This interaction creates recirculation of the hot gases into the missile base region, which is evident in Fig. 5b.

The influence of the missile body and base regions on the plume flow field is assessed by comparing solutions obtained with and without the missile body and base region included in the simulation. Static temperature contours and centerline static temperature profiles from these calculations are compared in Fig. 6. These results indicate that the plume expansion angles are larger and the

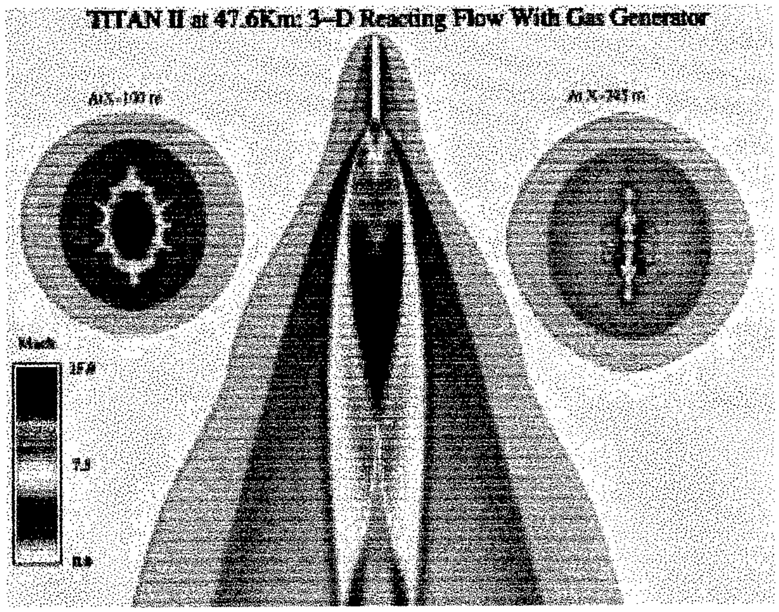
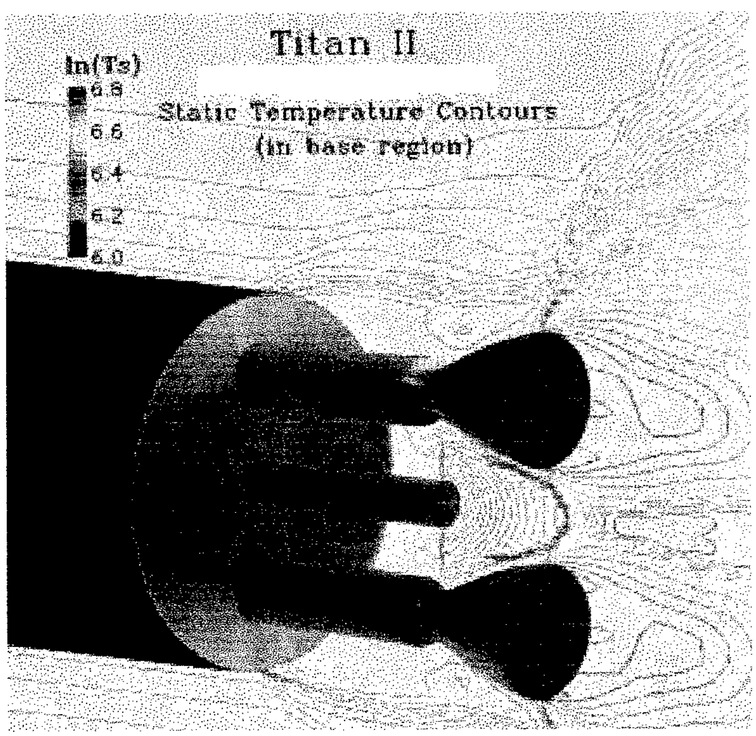


Fig. 4. Mach number contours for 3-D reacting flow with gas generator.

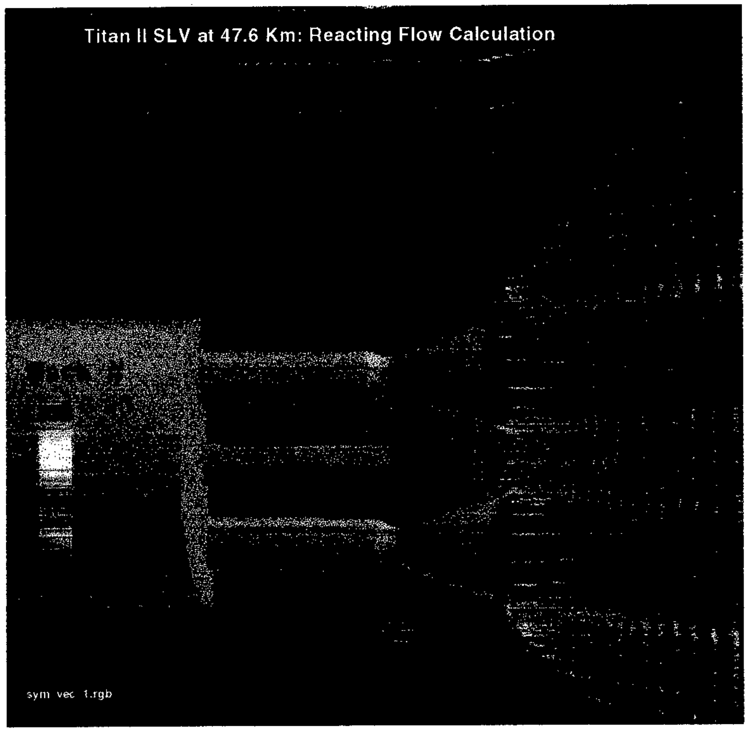


a. Static temperature contours

Fig. 5. Titan II SLV at 47.6 km reacting flow calculation near-field results.

overall plume width is larger if the missile body and base region are included in the computational domain. The shock reflection point was located 12 meters farther downstream in the solution including the body and base. It appears that the missile body effects are significant and influence the plume impingement shock location, the inviscid shock structure, and the plume shear layer development.

To assess the influence of the gas generator effluents, the nose-to-tail solution was repeated with and without consideration of the gas generator flow assuming frozen chemistry conditions. Comparison of these results indicates that the gas generator flow has a significant influence on the very near-field portion of the plume flow field, extending to approximately 1 meter downstream of the nozzle exit. The effect of the gas generator is not noticeable at further downstream locations. To assess the effect of the gas generator, Fig. 7 contrasts Mach number contours and radial Mach number profiles taken at 2- and 3-m axial locations downstream of the nozzle exit plane location with and without the gas generator flow. The gas generator flow region is apparent near the centerline. This solution assumed a frozen chemistry condition throughout the computational



b. Mach vectors
Fig. 5. Concluded.

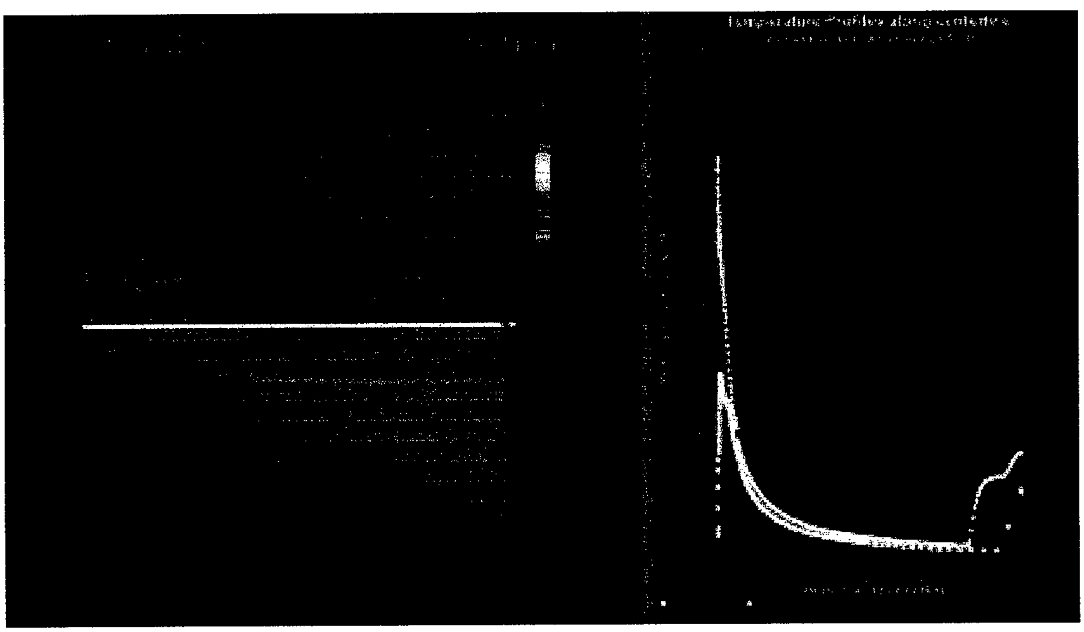


Fig. 6. Static temperature contours with and without missile body.

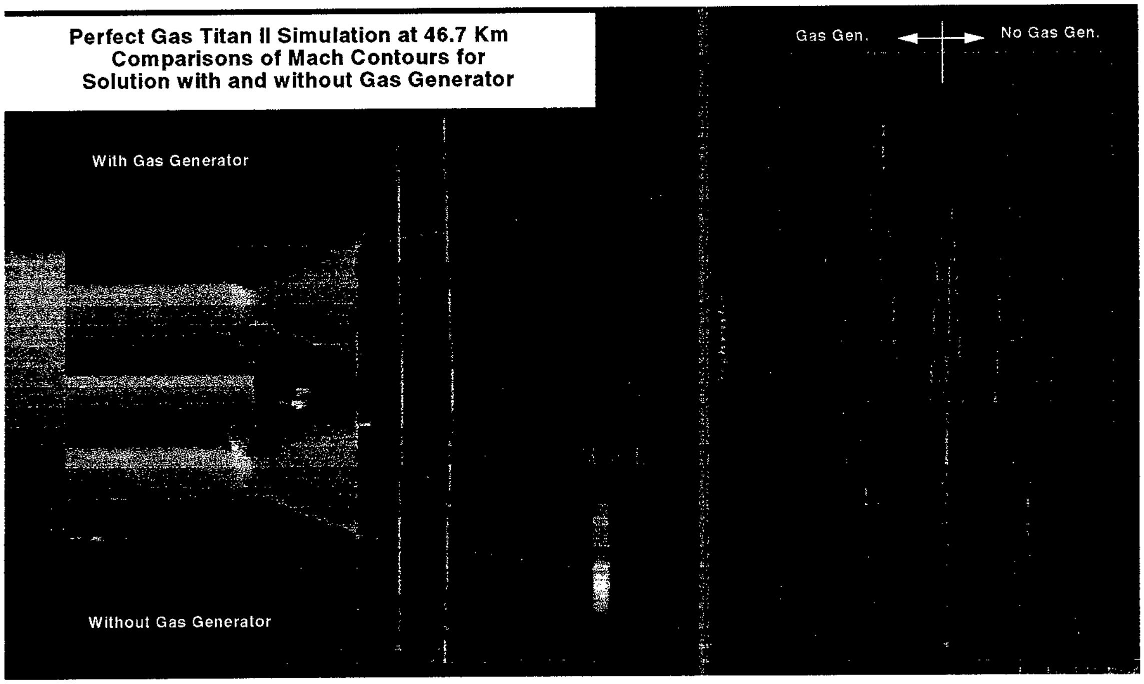
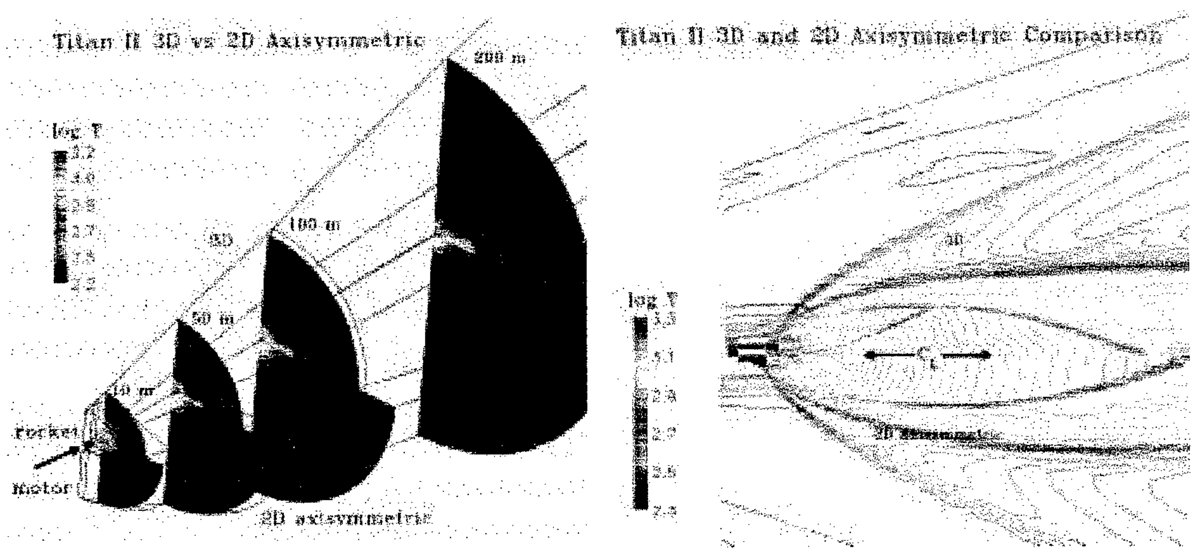


Fig. 7. Mach number comparison with gas generator versus without gas generator.

region. Comparison of this result with the reacting gas generator results indicates that in order to accurately simulate missile base heating kinetic chemistry is paramount. For reacting flow conditions, it is also expected that the influence of the fuel-rich gas generator flow would generate strong localized combustion zones extending to further downstream positions.

A comparison of the two-dimensional axisymmetric solution with the three-dimensional twin-nozzle solution was accomplished to determine the impact of the single equivalent nozzle assumption. This simplification is often used to approximate and simplify three-dimensional geometries. Figures 8a and 8b are static temperature contours and provide a comparison of the axisymmetric and the 3-D



a. 3-D plume (top) versus single equivalent nozzle (bottom)

b. 3-D (top) versus asisymmetric (bottom)

Fig. 8. Static temperature contour comparisons.

solutions. In both figures, the 3D solution is shown as the upper half of the contour plot and the axisymmetric solution as the lower portion of the figure. There are significant differences between the two solutions, including the overall plume size, asymmetric flow distributions, and the location of the shock reflection points. This illustrates that a single equivalent nozzle assumption provides some generalized insight concerning overall, nominal qualitative assessments; however, for detailed studies requiring accurate resolution of the spatial character of multinozzle plume flows, a three-dimensional calculation is required. The 3-D influence is further magnified when the body/base are included in the simulation.

Conclusions

A modified, three-dimensional GIFS computer program was applied extensively for this study, which focused on evaluating the three-dimensional effects of the missile body, missile base region, and the gas generator on the resulting plume exhaust flow-field simulation. The steady-state flow field resulting from a Titan II propulsion system operating at high-altitude (47.6 km) flight conditions was parametrically simulated in this numerical investigation. Previous studies investigated the effects resulting from startline nonuniformity, intranozzle spacing, turbulence, and finite-rate chemistry for the same Titan II twin-nozzle/plume propulsion configuration and operating conditions simulated in this study. Analysis of all the results, including the previous study, leads to the following conclusions:

1. Nozzle exit startline assumptions, radial profile versus one-dimensional uniform, had a significant effect on the immediate plume near-field calculation. The difference between the two startline assumptions became less significant as the axial distance downstream of the nozzle exit plane was increased. This significance of the observed sensitivity needs additional investigation to elucidate the effect of the starting conditions on the plume flow field, especially in the presence of strong afterburning and missile body/base flow-field interactions, which are expected to dominate lower altitude simulations.

2. Comparison of solution results contrasting laminar and turbulence stress models indicates that plume flow-field simulations are significantly influenced by the turbulence model. Laminar approximations will degrade further at lower altitudes where turbulent mixing becomes more dominant. In general, the turbulent approximation resulted in slightly increased temperatures throughout the flow domain.

3. The chemistry assumption (frozen versus reacting) did not influence the overall plume structure in this case. However, at lower-altitude conditions where afterburning occurs, the chemistry effect may be significant. In addition, the chemistry assumption will dramatically influence the heat-transfer implications in the missile base region and in the recirculation and impingement regions between multiple nozzles.

4. The intranozzle spacing distance has a significant impact on the barrel shock formation and reflection location, plume/plume impingement shock location, and shear layer development. Instances where these influences may impact flow-field predictions occur when nozzle gimballing cycles are considered.

5. A comparison of the two-dimensional and three-dimensional cases indicates that the three-dimensional effects are important in the near-field plume and diminish as the axial distance extends farther downstream from the nozzle exit plane. The single, equivalent nozzle approach should not be used to describe plume near-field flow characteristics where three-dimensional effects are predominant, and in the instances where 3-D features and spatial information are required as part of the flow-field description.

6. The missile body has a significant impact on the barrel shock reflection location, plume/plume impingement shock location, and development of the shear layer region. The shock layer initiated near the missile nose increased the temperature surrounding the missile body. This temperature increase is expected to influence the entire flow-field domain.

7. A comparison of solutions contrasting gas generator effects indicates that the gas generator

flow has a pronounced influence in the immediate plume near-field region and diminishes beyond approximately 1 meter downstream of the nozzle exit location. The extent of the effect is expected to be based on the gas generator mass flow. The gas generator flow is expected to contribute to the missile base heating and nozzle wall impingement heating, resulting in recirculation of hot gas in the region between the two nozzles. However, the current frozen chemistry solution did not adequately simulate the heating effect.

8. These analyses indicate that two-dimensional, axisymmetric, nonequilibrium analysis tools can provide general insight concerning generalized qualitative assessments of multiple plume flow-field phenomena. However, for detailed studies of complex flow-field phenomena and spatial generalizations, a more sophisticated three-dimensional calculation is required.

Furthermore, this study is intended to assess, understand, and quantify propulsion and plume physical phenomena in order to identify where simplified models can be used without introducing significant errors in conclusions that might be deduced from analysis of the flow-field simulation. Toward this objective, this study explored three-dimensional vehicle/base interactions with the plume flow. Also, to promote optimal utilization of computer resources, hybrid Navier-Stokes/ Parabolized Navier-Stokes methodologies have proven to be very efficient.

Acknowlegments

Acknowledgment is given to Mr. Roy Hilton for his valuable suggestions and technical support. Special gratitude is extended to Ms. Martha A. Simmons for her technical support and assistance in preparing this paper.

References

1. Shannan, Robert V. and Murray, Alvin, "Development of the KIVA-II CFD Code for Rocket Propulsion Applications," Tenth Workshop for CFD Applications in Rocket Propulsion, NASA Marshall Space Flight Center, April 28-30, 1992.

2. Ebrahimi, Houshang B., "Numerical Investigation on Multi-Plume Rocket Phenomenology," Joint Propulsion, AIAA 97-2942, Seattle, WA, July 6, 1997.

3. Dash S. M., and Thorpe, R. D., "Dash, Shock Capturing Model for One and Two-Phase Supersonic Exhaust Flow," *AIAA Journal*, Vol. 19, July 1981, pp. 842-851.

4. Dash, S. M., Wolf, D. E., Beddini, R. A. and Pergament, H. S., "Analysis of Two-Phase Flow Processes in Rocket Exhaust Plumes," *J. Spacecraft Rockets*, Vol. 22, No. 3, May-June 1985, pp. 367-380.

5. Ebrahimi, Houshang B., Levine, Jay, Kawasaki, Alan, "Numerical Investigation of Twin-Nozzle Rocket Plume Phenomena," AIAA-97-0264, 35th Aerospace Sciences Meeting and Exhibit, Reno, NV, January 6-10, 1997.

6. Holcomb, J. Eric, "Development of an Adaptive Grid Navier-Stokes Analysis Method for Rocket Base Flows," AIAA-88-2905, Boston, MA, July 1988.

7. Van Leer, B., "Flux-Vector Splitting for the Euler Equations," in lecture note in *Physics*, Vol. 170, Springer-Verlag, 1982.

8. Ebrahimi, Houshang B., "Validation Database for Propulsion Computational Fluid Dynamics," *Journal of Spacecraft and Rockets*, Vol. 34, Oct. 1997, pp. 642-650.

9. Nickerson, G. R., "Two-Dimensional Kinetics (TDK) Nozzle Performance Computer Program," NAS8-36863, March 1989.

10. Walters, W. R., Cinnella, D. C., and Holt, S. D., "Characteristic-Based Algorithms for Flows in Thermochemical Nonequilibrium," *AIAA Journal*, Vol. 30, No. 5, 1992, pp. 1304-1313.

11. *Titan II SLV Propulsion Handbook*, Gencorp, Aerojet Division, Sacramento, CA, 1992.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	January 1998	Technical Society Paper

4. TITLE AND SUBTITLE
Numerical Investigation of Twin-Nozzle Rocket Plume Phenomenology
AIAA Paper No. 98-0924

5. FUNDING NUMBERS
Job No. 4458

6. AUTHOR(S)
Ebrahimi, Houshang B.

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Arnold Engineering Development Center/DOT
Air Force Materiel Command
Arnold AFB, TN 37389-9011

8. PERFORMING ORGANIZATION REPORT NUMBER
1

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Arnold Engineering Development Center/DOT
Air Force Materiel Command
Arnold AFB, TN 37389-9011

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
Presented at the 36th Aerospace Sciences Meeting & Exhibit in Reno, NV.

12a. DISTRIBUTION AVAILABILITY STATEMENT
Approved for public release; distribution unlimited.

12b. DISTRIBUTION CODE
A

13. ABSTRACT *(Maximum 200 words)*
The Generalized Implicit Flow Solver (GIFS) computer program has been modified and applied for analysis of three-dimensional, reacting, two-phase flow simulation problems. The intent of the original GIFS development effort was to provide the Joint Army, Navy, NASA, Air Force (JANNAF) community with a standard computational methodology to simulate multiple nozzle/plume flow-field phenomena and other three-dimensional effects. The Van Leer Flux Splitting option has been successfully implemented into the existing GIFS model and provides a more robust solution scheme, making application of the model more reasonable for engineering applications. This paper is a continuation of the previous work and reports the significant results of parametric flow-field simulations resulting from a dual-nozzle propulsion system operating at a high-altitude flight condition. In support of this effort, four calculations of Titan II SLV flow fields have been completed to assess the effects of three-dimensionality, missile body effects, chemistry, and gas generator flow on simulated plume exhaust flow-field properties. These calculations indicate that three-dimensionality is always an important factor and could substantially influence the interpretation of the results. If three dimensional effects are oversimplified in the model, analyses of the spatial results can be misinterpreted and misapplied. The missile body effect can also generate three-dimensional influences affecting the Mach reflection location, the plume/plume impingement shock location, inviscid shock structure, and shear layer growth. The gas generator flow influences the very near-field simulation but is significantly dampened beyond approximately 1 meter downstream of the nozzle exit (nonreacting). If missile base heating, recirculation effects, or nozzle

14. SUBJECT TERMS
multi-plume, solid-propellant rocket motor, phenomenology, flow field

15. NUMBER OF PAGES
12

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	UL